SolidWorks and Vacuum Forming and Part Finishing Oh my!
Logistics and Reminders

Design consultations!

Next Wednesday during lecture

Two separate models:

a works-like

+ 

a works-like or looks-like

At least one model is play-able for play testing (April 21st)
Tutorials and Demos

Mold Making and Casting with a Reynolds Materials Rep
Tuesday 2pm-4pm in the PDL

Introduction to Electronics with Ben Nahill
Tuesday 7pm-8pm in 3-370
SolidWorks
and
Vacuum Forming
and
Part Finishing
Oh my!
SolidWorks Basics
When to use CAD?

quick prototypes

organic shapes

detailed planning

calculating
digital fabrication
SolidWorks Basics
SolidWorks Navigation

- Command Bar
- Feature Tree
- Model Window
Simple Box

Select the “Front” plane
Create a new sketch
Create a “Center Rectangle” from the origin
Smart Dimension the length and width in inches
Change the View

Change the view to “Trimetric”

Use middle-click and drag to free rotate
Extrude

Extrude the sketch

Choose “Mid Plane”, and set to 3.125"
Modifying the Sketch

Delete the extruded feature (we’ll come back to that later)

Edit the sketch, draw a horizontal line below the box

Set the line as “for construction”

Dimension the line 0.275" below the box
Sketching the Wheel

Draw a circle, connecting the edge to the construction line so it is tangent
Dimension the circle 1" away from the front
Dimension the diameter as 42mm (notice it change to inches)
New Sketch

Create a new sketch on the “Front” plane
Use “Convert Entities”
Select the rectangle to convert (this brings it to the new sketch)
Extrude the box as before, with “Mid Plane” & 3.125"
Reference Plane

In the top menu, choose:

“Insert > Reference Geometry > Plane”

Set the plane 0.10" away from the box face
New Sketch

Create a new sketch on the new reference plane
Use “Convert Entities” again
Select the circle to bring it onto this new sketch
Extrude the wheel

Extrude the wheel

Set the extrude as “Blind” and to 0.60"

Uncheck the box for “Merge result”

really, do it!
Round the Corners

Select the “Fillet” tool
Select the edges of the box to fillet
Set the radius to 0.25"
Round the Edges of the Wheels

Select the “Fillet” tool
Select the edges of the wheel to fillet
Set the radius to 0.125"
Mirror the Wheels

Select the “Mirror” tool

Choose to mirror about the “Front” plane

Go to “Bodies to Mirror”, then click on the wheel in the model window (you can check preview to see what will happen)
Mirror the Wheels

Select the “Mirror” tool again

Choose to mirror about the “Right” plane

Go to “Bodies to Mirror”, then click on the two wheels in the model window
Sketch the Canopy

Edit the original sketch on the “Front” plane
Sketch a line at an angle on the left side
Draw an arc tangent to the angled line and ending on the right, at the top of the car
Dimension from the edges: 1.70", 2.00", and angle of 25°
(Also, try dragging around the point on the left to change the shape of the canopy)
Extrude the Canopy

Create a new sketch on the “Front” plane
Use “Convert Entities” to bring over the canopy sketch
Draw a line to close the canopy shape
Extrude the canopy using “Mid Plane” and set to 2.25"
Uncheck the box for “Merge result”
Round the Edges of the Canopy

Make sure you unchecked the box for “Merge result” from the previous step

Fillet the top edges of the canopy to 0.35"
Hide the Body

Right click on the feature for the car body, and choose “Hide” to make it invisible.

Rotate the view to see the underside of the canopy.

Right click to “Hide”
**Shell the Canopy**

Select the “Shell” tool

Set the shell thickness to 0.10"

(notice how this hollows out the canopy)
Glass Canopy

Flip the car back around using “Isometric”
Make the car body visible again by right-clicking “Edit the Appearance” to make it more realistic
Other things to try...

Try modifying the original rectangle for the car body to make it better match the Automoblox cars.

What happens when you change the dimension for the “ground clearance” construction line?

Try adding a spoiler or other detail.

Make axles and corresponding holes for the wheels.
Create new parts from existing bodies

Expand the “Solid Bodies” folder

Right click on the “canopy” body

Choose “Insert into New Part…”
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